

ANSYS Procedure for 2D Field Analysis and Iron Saturation Study

J. Moeller

*Technical Division, Development & Test Department
Fermilab, Batavia, IL - 60510*

Synopsis

This note contains the following items:

- A brief conceptual background on the 2D saturation problem.
- Information on starting the ANSYS 5.5 program at Fermilab.
- Instructions for modeling a magnetic dipole in ANSYS.
- Instructions for static solution of model at multiple excitation currents.
- A procedure for harmonic analysis of extracted field data.
- A procedure for plotting a median plane field variation curve.
- Instructions for adding holes to the iron yoke for field correction.

1. Background

It is understood that the undesirable sextupole harmonic observed in the field of *cosine-q* dipole magnets is caused partially due to saturation effects of the iron yoke. At high field, flux saturates the iron, becoming more concentrated towards the top and bottom of the aperture than it does near the sides. It is this uneven distribution of flux around the aperture that adds to the sextupole component in the field. One way to reduce this effect is to alter the geometry of the iron yoke in a way that promotes a more uniform flux distribution around the aperture. This can be done by introducing holes into the iron at certain locations to hinder the dispersion of flux near the sides of the aperture.

The purpose of this note is to describe a procedure used to model the problem described above using ANSYS. It also includes information on how to extract and process harmonic information derived from the simulation using the ROXIE-TRICOF procedure.

2. Starting ANSYS 5.5

In order to use ANSYS 5.5 at Fermilab, you need to have login access to a UNIX node that it resides on. You also need to add the following line to your .cshrc file.

```
setenv ANSYS_ELMHOST @cadwhs05.fnal.gov
```

If you are using a PC to access the node via Exceed, be sure to set the DISPLAY environment variable to your PC's IP name as follows.

```
setenv DISPLAY mypcname.fnal.gov:0
```

Once the environment variables are set appropriately, ANSYS can be invoked graphically with the Emag3D package with the following command.

```
/ansys55/bin/ansys55 -g -p ANE3
```

3. ANSYS Modeling

There are quite a few steps to consider when assembling your ANSYS model. The following list outlines the basic steps that need to be taken.

- Begin with a coil geometry optimized by the ROXIE program. Its output file will contain a list of point coordinates representing the four corners of each conductor turn. Use these as keypoints from which to construct the conductor turn areas.
- Add insulation areas between the conductor turns as well as between the layers and on the outer edges. This will require you to add corner points offset from the edges of the coil by the insulation thickness. To create the areas, select all keypoints and lines involved in the area, and use either the `a` or `a1` commands.
- Add all other geometry. This includes an area for the aperture vacuum, the collar, iron yoke, and outside air beyond. Arc lines can be drawn between two points by simply changing into the global cylindrical coordinate system with `csys,1`. Also use cylindrical coordinates when defining the areas bounded by such arcs to ensure the proper shape.
- After definition of all areas, be sure to assign the appropriate material attributes to the various areas. For magnetic analysis, assign a relative permeability of 1 to all materials except iron. For the iron, you can use either a specific constant value or specify a B-H curve. It is especially important to have an accurate iron permeability value for this type of analysis.
- Finally, the model must be meshed. Line-element divisions in key areas should be modified using the `lesize` command. For the most part, default mesh patterns will do for this analysis, although the element size should be somewhat smaller for the collar/coil region than the yoke. Once you are satisfied with the model definition and mesh, you are ready to begin solution.

4. ANSYS Solution

Once the dipole model is defined and meshed to satisfaction, it is ready to be excited. To do this, simply add a current density body load to the conductor elements. Since you will want to get an idea of the harmonic behavior vs. current, it is best to apply several load steps of various currents within the range of the power source. The following snippet of ANSYS input code shows how two load steps can be set up.

```
c***** load step 1
/tit, 1 kA with 70mm/30mm/16mm holes      ! assign a new title
esel,s,type,,2                             ! select conductor elements
bfe,all,js,3,cur_den1                      ! assign current dens. #1
allsel
nsub,1                                     ! set # of substeps
neqi,25                                    ! set # of equil. iterations
lnsrch,on                                  ! set line search on
solve                                       ! solve for this load step
c***** load step 2
/tit, 5 kA with 70mm/30mm/16mm holes      ! assign a new title
esel,s,type,,2                             ! select conductor elements
bfe,all,js,3,cur_den2                      ! assign current dens. #2
solve                                       ! solve for this load step
```

Note that in the above code, the current densities were previously defined as the parameters `cur_den1` and `cur_den2`. These were determined by dividing the desired currents of 1 kA and 5 kA by the cross-sectional area of an individual conductor turn.

ANSYS will save all results from the multiple load steps into a single file `*.rmg` after the solution is done. You will use the postprocessor to look at the results and extract data.

5. Extraction of Harmonics

In order to calculate the harmonic multipole components of the field, you need to extract raw field data from the model and perform a Fourier analysis. For this we use CERN's TRICOF procedure that has been extracted from ROXIE by V. Kashikhin. Input for this program should be a list of radial B_r components sampled along an arc of radius r_0 in the magnet bore. This data can be extracted using a PATH operation and saved as an ASCII file. After the file is formatted properly, TRICOF can read it and calculate harmonic results.

The general procedure for extracting harmonics is as follows.

- First, you will need to locate the two end nodes from which to create a circular arc path at a radius of r_0 . Pull up a line plot of your model and zoom in on the aperture. Use graphical selection to select a bunch of nodes near $(r_0, 0)$ and $(0, r_0)$ and then list all the nodes you selected. Look for the two nodes that have the closest coordinate value to what you are looking for and jot down their corresponding node numbers. You will need these to define the path.
- Now you need to define the path. To create a curved path between the two nodes, you need to first get into the cylindrical display coordinate system by typing `dsys,1`. You will also want the results to be in cylindrical so next type `rsys,1`. Now type `lpath,[node #1],[node #2]` to define the path. Use the node numbers you found for nodes #1 and #2 with node #1 being the one on the x-axis.
- Next, extract the field data. Type `pdef,[name],b,x` with `[name]` being whatever name you want to give to the data set produced. The `b` argument specifies magnetic flux density, and the `x` specifies radial component in the cylindrical system. Next, type `prpath,[name]` to list the data. Click Save As and give a filename so that you can edit the data for input to TRICOF.
- If you want to see what the path looked like graphically, type the following sequence of commands: `dsys :: /pbc,path,1 :: allsel :: nplot`. You should verify that the r_0 arc path was created properly.
- Repeat the above steps for each of the multiple load steps. When you are done, you should have a bunch of files containing B_r field data for different currents. Before you can run the TRICOF routine on them, you need to first edit out extraneous data.
- Open up the files in a text editor such as WinEdit and delete all the interspersed text among the numbers. Then use column block to remove the entire first column that is headed by an S. While you are at it, delete all the leading minus signs off the data in the second column. The finished files should contain only the unsigned B_r data. Save the edited files in the directory where the TRICOF executable resides.

- Run the TRICOF executable on the files. The executable itself is called Harm.exe. It will ask for a filename and then dump out harmonic results. These results are automatically saved in Harm.txt. From here, you simply need to rerun the program on each of your files and move the data from Harm.txt into a suitable presentation format within EXCEL. A graph of harmonics versus current is the most ideal form for this type of analysis.

6. Plotting Field on Median Plane

To obtain a plot of the field variation on the median plane of the dipole, the procedure is similar to that of the harmonic extraction. The main difference is in the path definition. You need to obtain node numbers for the central node and the node at the extreme x-value in the aperture. Then set `dsys` and `rsys` to Cartesian coordinates and create the path as a horizontal segment along the x-axis of the aperture. Extract the y-component of the B-field using `pdef,[name],b,y`. Once again save the `prpath` file and edit out the extraneous interspersed text. Keep the S-values though, as you will need them to graph as the independent distance variable. Import the raw data into EXCEL and use formula operations to normalize the field data by the central value. Set up an x-y scatter plot using the S-values for your x-values and the normalized B_y/B_0 field values for the y-values. You will now have a curve showing the variation of the field in the aperture.

7. Adding Holes To Iron Yoke

To introduce holes into the iron, one only has to modify the geometry a bit before solution. The easiest way to do so is to draw circular areas where the holes are to be and then subtract them from the iron area. The following snippet of ANSYS code shows an example of how this can be done.

```
cyl4,0.10,0.11,0.035,0,0,360,0      ! create 1st hole (#67)
cyl4,0.08,0.04,0.015,0,0,360,0      ! create 2nd hole (#68)
cyl4,0.08,0.01,0.008,0,0,360,0      ! create 3rd hole (#69)
aatt,1,1,1                          ! attribute (air)

( commands to draw the solid iron here )

allsel
asba,70,67,,delete,keep              ! subtract 1st hole
asba,71,68,,delete,keep              ! subtract 2nd hole
asba,70,69,,delete,keep              ! subtract 3rd hole
asel,u,area,,all
asel,s,area,,71                      ! select only iron (#71)
aatt,6,6,6                          ! attribute (iron)
```

The first three commands draw circular areas at certain locations and radii. The commands to draw the solid iron area would immediately follow the creation of the circles. Next, area subtraction commands are used to remove sections of the iron where

the holes are. These commands must be given explicit numbers for the existing areas involved. Since ANSYS reassigns area numbers when areas are rearranged or deleted, one must keep an eye on the actual area numbers before and after each command when doing it sequentially like this. For example, the first `asba` command says to subtract area 67 from area 70 and to remove area 70. The output of this command has now made the cut iron area 71. Thus, the second command must specify area 71 as the iron area to be cut from. Since the second command deletes area 71, the iron once again becomes area 70 for the third command. Once the final iron area is made, it is selected as area 71 and given the appropriate attribute set for iron material.

Once you have the input file set up correctly for holes, you can solve the model for harmonics as described above and then fiddle with the hole configuration. To do this, all you need to do is change the parameters for the `cyl4` commands defining the holes.

Acknowledgements

Bob Wands and Ang Lee for offering their expert advice on using ANSYS.
Vadim Kashikhin for extracting the TRICOF procedure from ROXIE for independent use.